

CFD ANALYSIS ON RADIAL COOLING OF GAS TURBINE BLADE

HARSHA D A

PG Student, Department of Aerospace Propulsion Technology, VTU, CPGS, Bengaluru Region VIAT, Muddenahalli, India harshaubdtmech@gmail.com

DR. YOGANANDA A

Associate Professor, Department of APTVTU, CPGS, Bengaluru Region VIAT, Muddenahalli, India

ABSTRACT

Gas turbines are extensively used for air craft propulsion, land based power generation and industrial applications. Thermal efficiency of gas turbine improved by increasing turbine rotor inlet temperature. The current rotor inlet temperature in advanced gas turbine is for above the melting point of blade material. A sophisticated cooling scheme must be developed for continuous safe operation of gas turbines with high performance. Gas turbines are cooled externally and internally. Several methods have been suggested for the cooling of blades and vanes. The techniques that involve to cool the blades and vanes by using cooling methods is to have radial holes to pass high velocity cooling air along the blade span. In this thesis, a turbine blade is designed and modelled in CATIA v5 and Icem CFD software. The turbine blades are designed using cooling holes. The turbine blade is designed with 12 holes. CFD analysis is done to determine the pressure distribution, velocity, temperature distribution and heat transfer rate by applying the inlet velocities. Thermal and Structural analysis is done to determine the heat transfer rates and strength of the blade. The present used material for blade is chromium steel. In this thesis, it is replaced with Titanium aluminium alloy. The better material for blade is analyzed.

KEYWORDS: CFD Analysis, Thermal and Structural analysis, Ansys V15, Radial cooling.

INTRODUCTION

With the advent in Gas turbine technology, its usage as a prime mover has become prominent, since last few decades. One of the most important applications of gas turbines is in power generation, though it has been in use for aircraft propulsion since long time.

The efficiency and power output of gas turbine plants is dependent on the maximum temperatures attained in the cycle. Advanced gas turbine engines operate at high temperatures (1200–1500°C) to improve thermal efficiency and power output. With the increase in temperatures of gases, the heat transferred to the blades will also increase appreciably resulting in their thermal failure. With the existing materials, it is impossible to go for higher temperatures. Taking into account the metallurgical constraints, it is necessary to provide cooling arrangement for turbine blades to keep their metal temperature within allowable limits. Therefore, developments in turbine cooling technology play a critical role in increasing the thermal efficiency and power output of advanced gas turbines.

Brahmaiah and Lava Kumar investigated the heat transfer analysis of gas turbine with four different models consisting of blade with without holes and blades with varying number of holes (5, 9 & 13). The analysis was carried out using commercial CFD software FLUENT (a turbulence realizable $k-\epsilon$ model with enhanced wall treatment) has been used. On evaluating the graphs drawn for total heat transfer rate and temperature distribution, the blade with 13 holes is considered as optimum. Steady state thermal and structural analysis was carried out using ANSYS software with different blade materials of Chromium steel and Inconel 718. While comparing these materials Inconel 718 is better thermal properties and induced stresses are lesser than the Chromium steel.

Chandrakant et al. compared the performance of helicoidal ducted blade cooling with turbulator of different geometric proportion. It was found from analysis that there was significant improvement in cooling characteristics for turbine blade with turbulator geometry having larger e/D ratio. Also it was found from his analysis, performance has been vastly improved for greater thickness of turbulator geometry. e = radial thickness of turbulator rib D = outer diameter of helicoidally duct e/D is non dimensional ratio chosen as parametric variable for different configurations of rib geometries.

MATHEMATICAL FORMULATION

2.1 The physical model Leading Edge Gas Turbine Blade.

X	Y	Z
48.5	0.5	0
45	3.95	0
38.2	8.77	0
26	13.6	0
21.1	14.9	0
16.18	15.5	0
11.2	14.5	0
6.18	12.4	0
3.2	13.5	0
2.6	17.3	0
5.85	21.5	0
10	25	0
14.8	26.6	0
22.9	25.3	0
24.5	24.7223	0
28	23	0
33.4	19.5	0
38	15.3	0
42	10.9	0
45.4	6	0
48.5	0.5	0

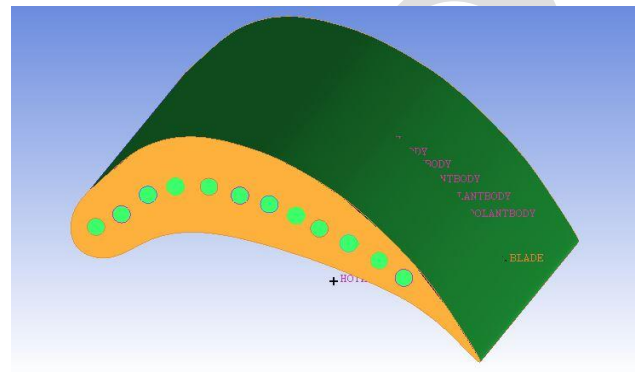


Fig.1 Create blade model with 12 holes

3. MESHING

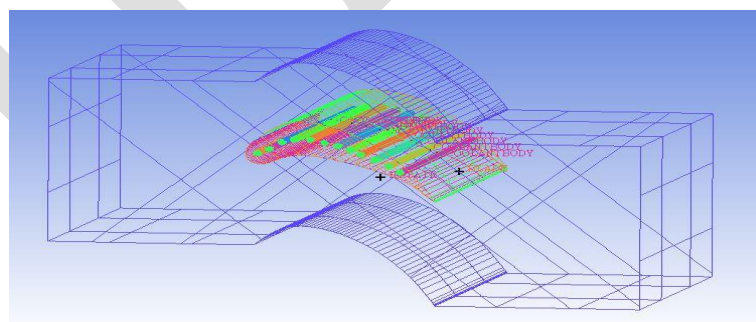


Fig.2 Geometry model with boundary walls.

- The model created using CATIA V5 and ICEM CFD software.
- The whole model is divided into different parts namely inlet, pressure outlet, wall and axis.
- Global Mesh parameters are defined which gives information regarding type of mesh. The global element seed size, part parameters are setup and mesh is computed which gives the mesh information regarding total number of elements.
- An unstructured hexahedral mesh is generated in order to perform computations with the Octree approach. And unstructured mesh to blade because of the curved shape structure after setting up part parameters for various parts, a mesh is generated with nearly 3093582 elements

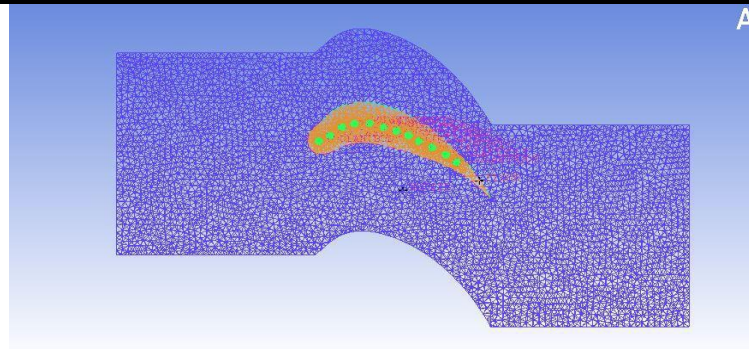


Fig.3 Meshed Model

BOUNDARY CONDITIONS

The steps followed for the solver setup is open the Setup option of ANSYS fluent project. Select Solver setup > General > among solver settings choose the pressure based solver. CFD fluent consist of two solvers for solving the CFD process i.e. pressure based and density based. Here we chooses pressure based; this is because the fluid under consideration is air at subsonic speed, where the compressibility effects are small and can be neglected. Choose the velocity formulation as absolute and time option as steady state.

This is because flow is considered to have constant properties with respect to time. Select Solver setup > Models > Choose energy equation on. Turbine blade analysis is aiming mainly at the temperature distribution over the turbine blade. To solve the heat transfer problems the basic energy equation has to be used and this can be used by switching on the energy equation. Select the Realizable K – epsilon model with enhance wall treatment as the viscous model. This is because the hot air velocity is sufficient to make the flow turbulence over the airfoil.

The Standard k–ε is a well-established model capable of resolving through the boundary layer .Select solver setup > Materials > select fluid and click edit. Edit the properties of the fluid with the properties given the table click change create. Select solver setup > Materials > select solid and click edit. Edit the properties of the solid with the properties click change create. Select Cell zone conditions and choose the appropriate materials for the zones like blade, hot air and cold air.

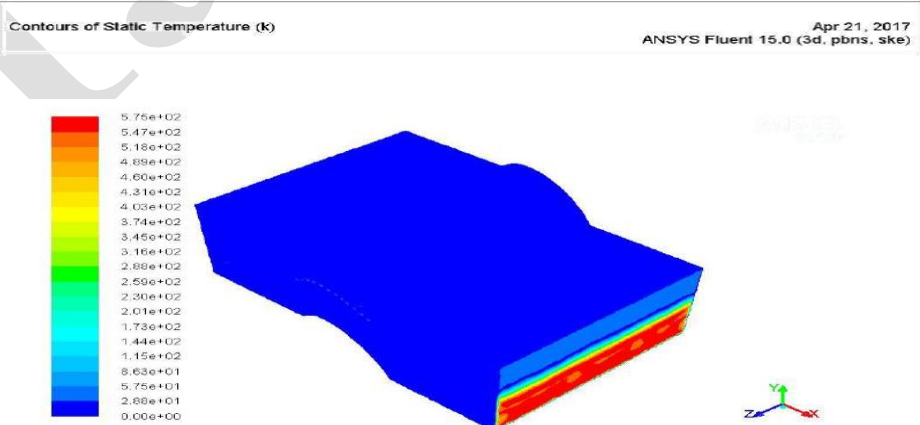
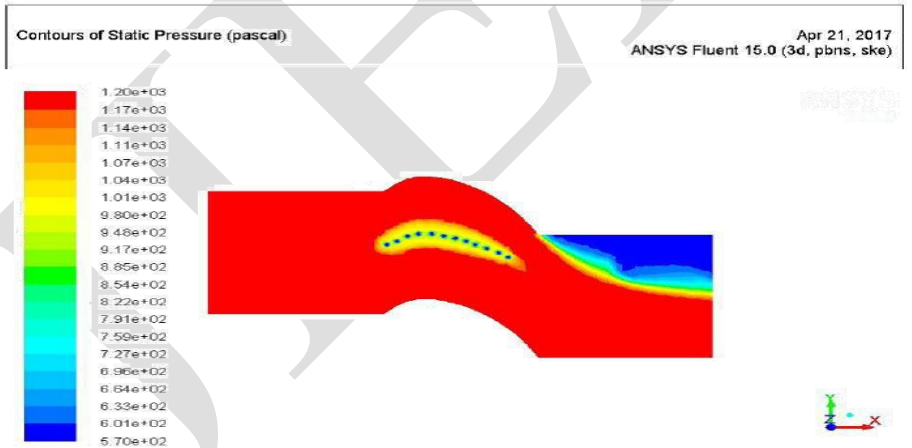
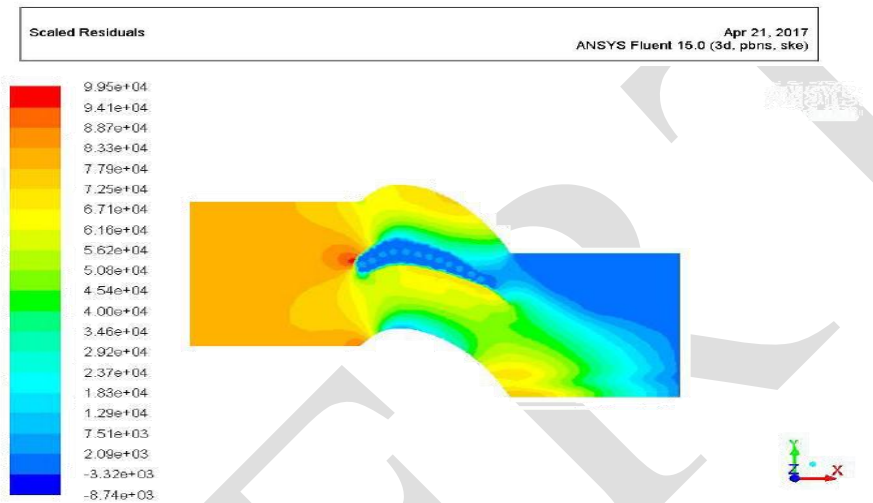
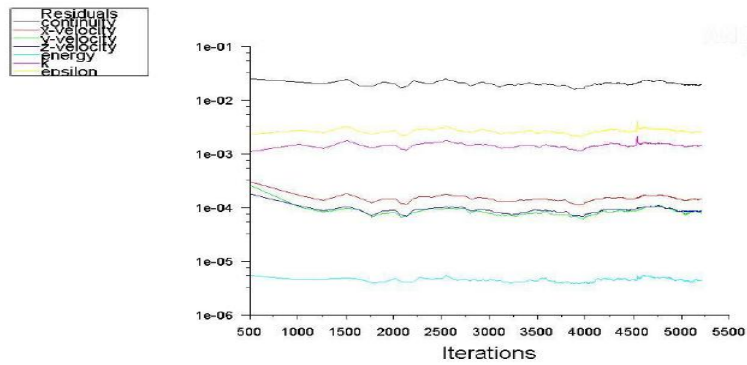
Select laminar for the zones of cold air. Click the operating conditions and verify that the operating pressure is 101325 Pascal. This is because we are assuming that this process is simulated at the atmospheric pressure at sea level. Select the boundary conditions and define the boundaries with appropriate input data According to our CFD model the hot air with temperature of 1200 K entering at a velocity of 265 m/s .Similarly the cold air at room temperature of 573 K is entering at a velocity of 30 m/s

In the solution method choose pressure velocity coupling scheme as SIMPLE. , SIMPLE is the acronym of semi implicit method for pressure linked equations.Choose momentum, turbulent kinetic energy and turbulent dissipation rate and energy as second order upwind. Second order upwind is the best discretization method for this project since the mesh is made with tetrahedral elements and the flow is not aligned with the grid. Select the monitors > residuals. Click edit. In the dialogue box that appears edit the convergence criteria of continuity, x velocity, y velocity, z velocity, energy, k, epsilon as 1×10^{-6} . Close the dialogue box. In the solution initialization select the hybrid initialization and click initialize.

MATERIALS	HOT-AIR	COLD-AIR	TITANIUM	SUPER ALLOY	ALUMINIUM (AD)
DENSITY	0.615	INCOMPRESSIBLE IDEAL GAS	4850	8190	2719
SPECIFIC HEAT (J/KG-K)	1047	1164	544.25	586	871
THERMAL CONDUCTIVITY (W/MK)	0.04605	0.074025	7.44	25	202.4
VISCOSITY	2.971e-05	4.5505e-05			

Table 1 Physical Property Details of Fluid Used In CFD Analysis

RESULTS AND DISCUSSIONS



CONCLUSION

In this work, a turbine blade is designed and modelled in CATIA v5 software. The turbine blades are designed using cooling holes. The turbine blade is designed with 12 holes. The present used material for blade is Titanium Alloy.

Thermal and Structural analysis is done By observing the CFD analysis results, the velocity of main stream(hot air) is decreased from 12 holes .But the value of pressure is slightly increased. The total heat transfer rate is maximum and the temperature of the leading edge is minimum for the blade consisting of 12 holes for titanium aluminum alloy

The temperature of the surface of blade with 12 holes for chromium is minimum. It is found that the temperature leaving the trailing edge is low leading to decrease in thermal efficiency of gas turbine. Result showed that heat transfer coefficient and Nusselt number on the surface of holes are nearly constant for all blade materials with different numbers of holes also the heat transfer coefficient is high at entrance region.

REFERENCES

- 1) A. Hasanpour, M. Farhadi and H.R. Ashorynejad , *Hole Configuration Effect on Turbine Blade Cooling, World Academy of Science, Engineering and Technology vol 49 2013.*
- 2) K Hari Brahmaiah, M. Lava Kumar, *Heat Transfer Analysis of Gas Turbine Blade through Cooling Holes , International Journal of Computational Engineering Research (IJCER) 2014.*
- 3) Chandrakant R Kini , *Computational Conjugate Heat Transfer Analysis of HP Stage Turbine Blade Cooling Effect of Turbulator Geometry in Helicoidal Cooling Duct , World Academy of Science, Engineering and Technology Vol:6 2012*
- 4) B. Deepanraj, *Theoretical Analysis of Gas Turbine Blade By Finite Element Method ,Scientific World, Vol. 9, No. 9, July 2011*
- 5) G Narasa Raju , *Steady State Thermal & Structural Analysis of Gas Turbine Blade Cooling System, International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 1, January- 2013*